

TAIYO YUDEN Component Library for Cadence PSpice (Temperature/DC Bias Model)

- Installation manual -

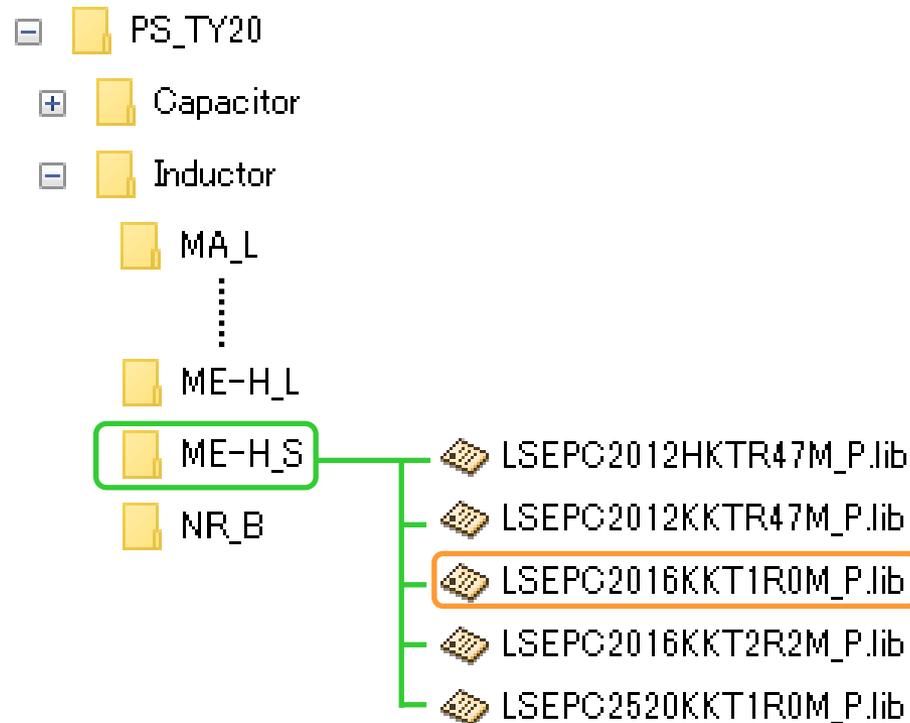
Contents

- * **How to install Component Library (P3)**
- * **How to use Component Library (P4-P5)**
- * **How to use Component Library on OrCAD (P6-P15)**

How to install Component Library

Step 1. Unzip “PS_TY**.zip”.

Step 2. Copy the netlist file(.lib) you would like to use to any folder you like.



How to use Component Library

Step 1. Describe the library in the netlist.

Step 2. Add the ambient temperature after the described library.

(In case of writing no parameter, 25C is set as the ambient temperature.)

netlist example

Step 1

```
Sample.cir - メモ帳
ファイル(F) 編集(E) 書式(O) 表示(V) ヘルプ(H)
I1 1 0 DC 0.3 AC 0.01
X1 1 0 LSEPC2016KKT1ROM PARAMS: Temperature=85
.ac oct 100 1MegHz 3GHz
.lib LSEPC2016KKT1ROM P.lib
.options numdgt=9
.print AC VR(1) VI(1)
.end
```

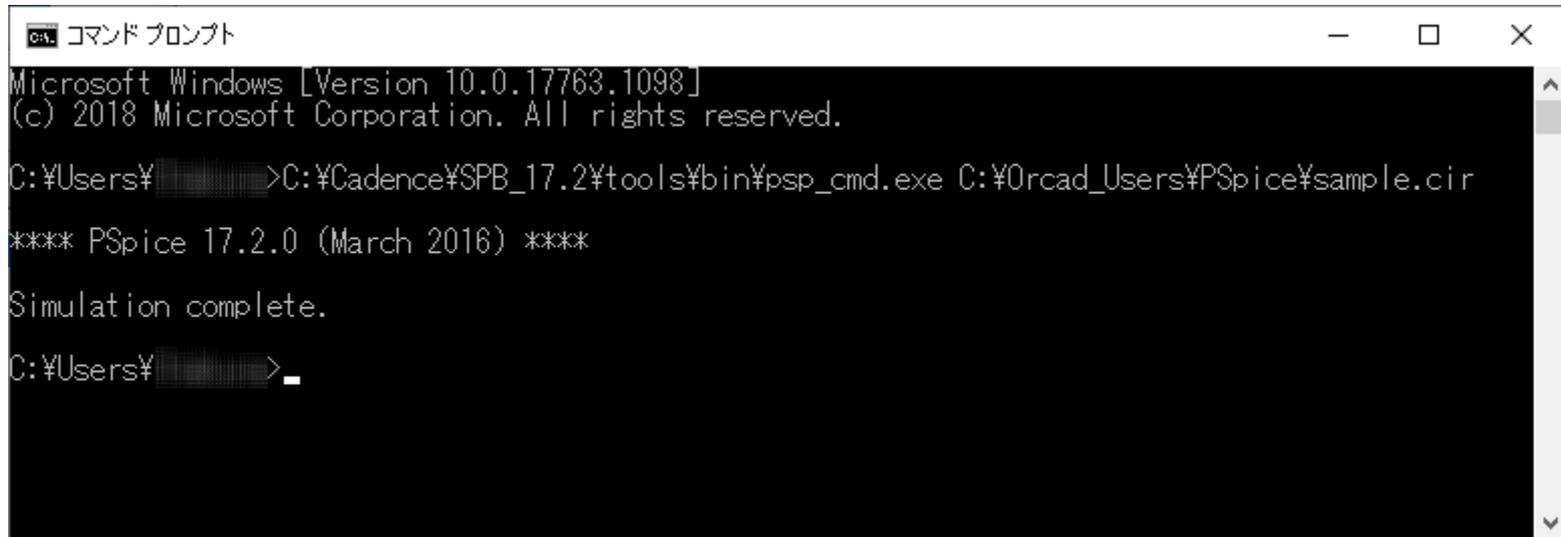
Step 2

***1** Refer to the PSpice manual for the description of the netlist.

***2** The ambient temperature here only works for the described library, not for the whole circuit.

How to use Component Library

Step 3. Perform the simulation from the command line such as windows command prompt.



```
CAV コマンド プロンプト
Microsoft Windows [Version 10.0.17763.1098]
(c) 2018 Microsoft Corporation. All rights reserved.

C:¥Users¥ [redacted] >C:¥Cadence¥SPB_17.2¥tools¥bin¥psp_cmd.exe C:¥0rcad_Users¥PSpice¥sample.cir

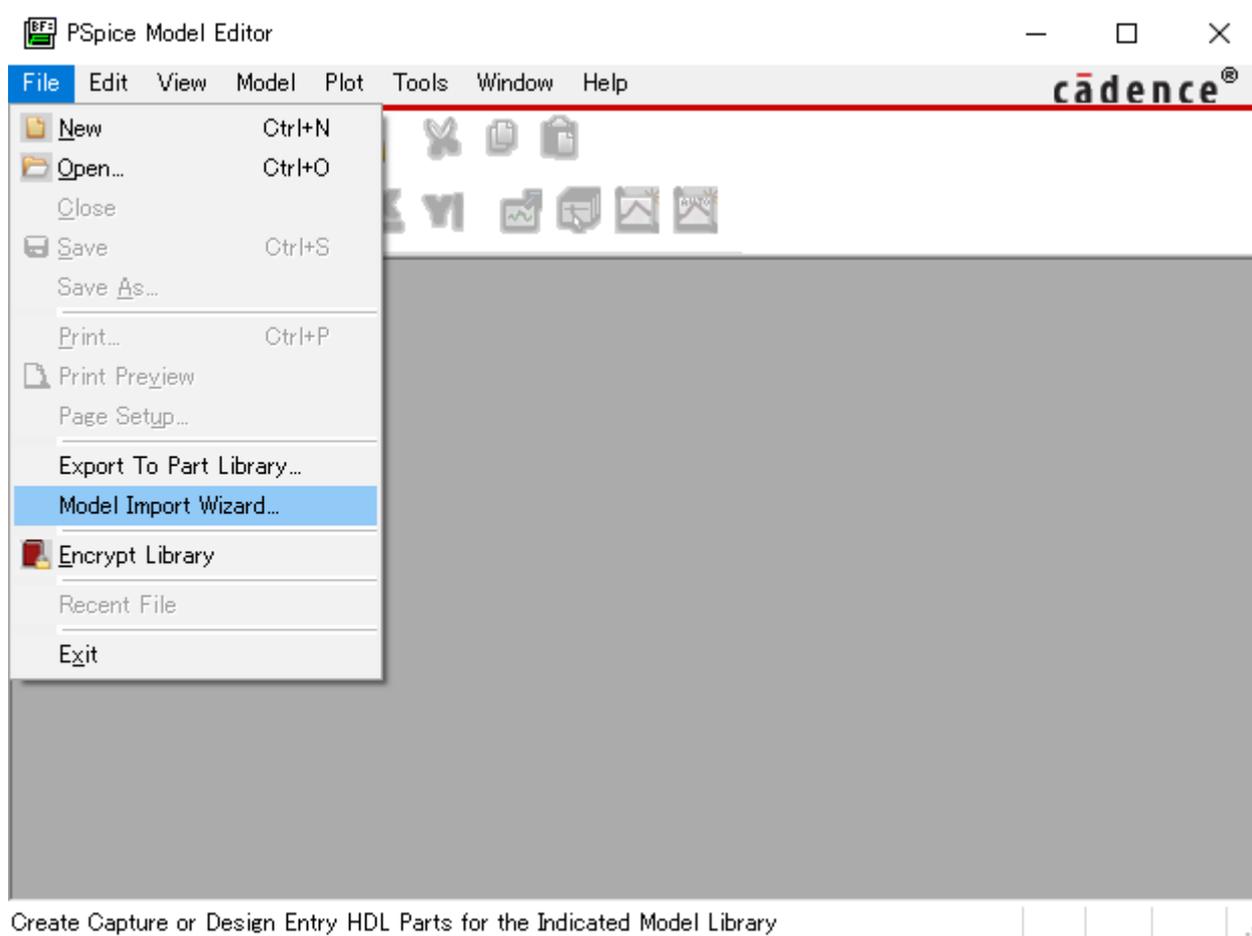
***** PSpice 17.2.0 (March 2016) *****

Simulation complete.

C:¥Users¥ [redacted] >_
```

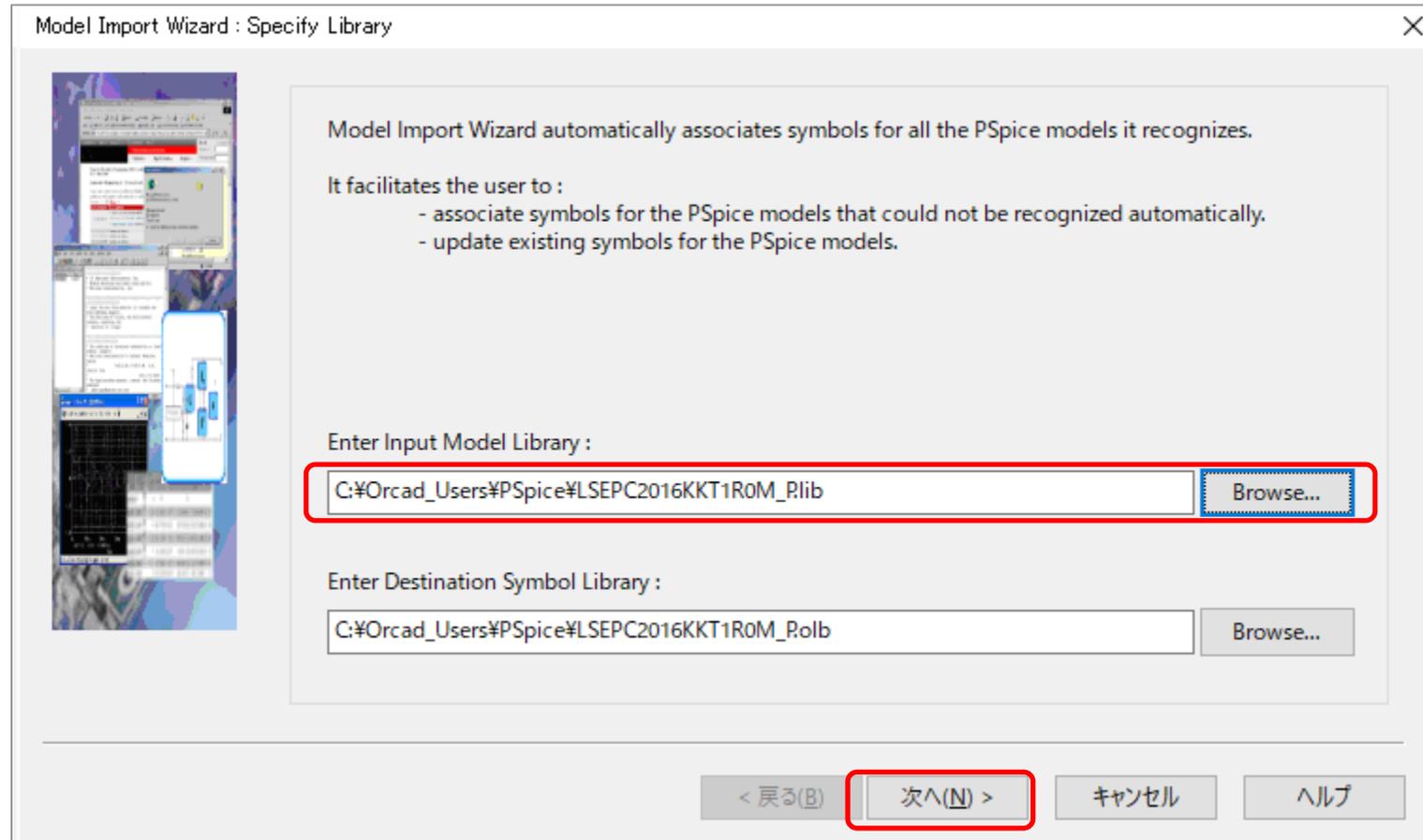
How to use Component Library on OrCAD

Step 1. Launch PSpice Model Editor, then select Model Import Wizard.



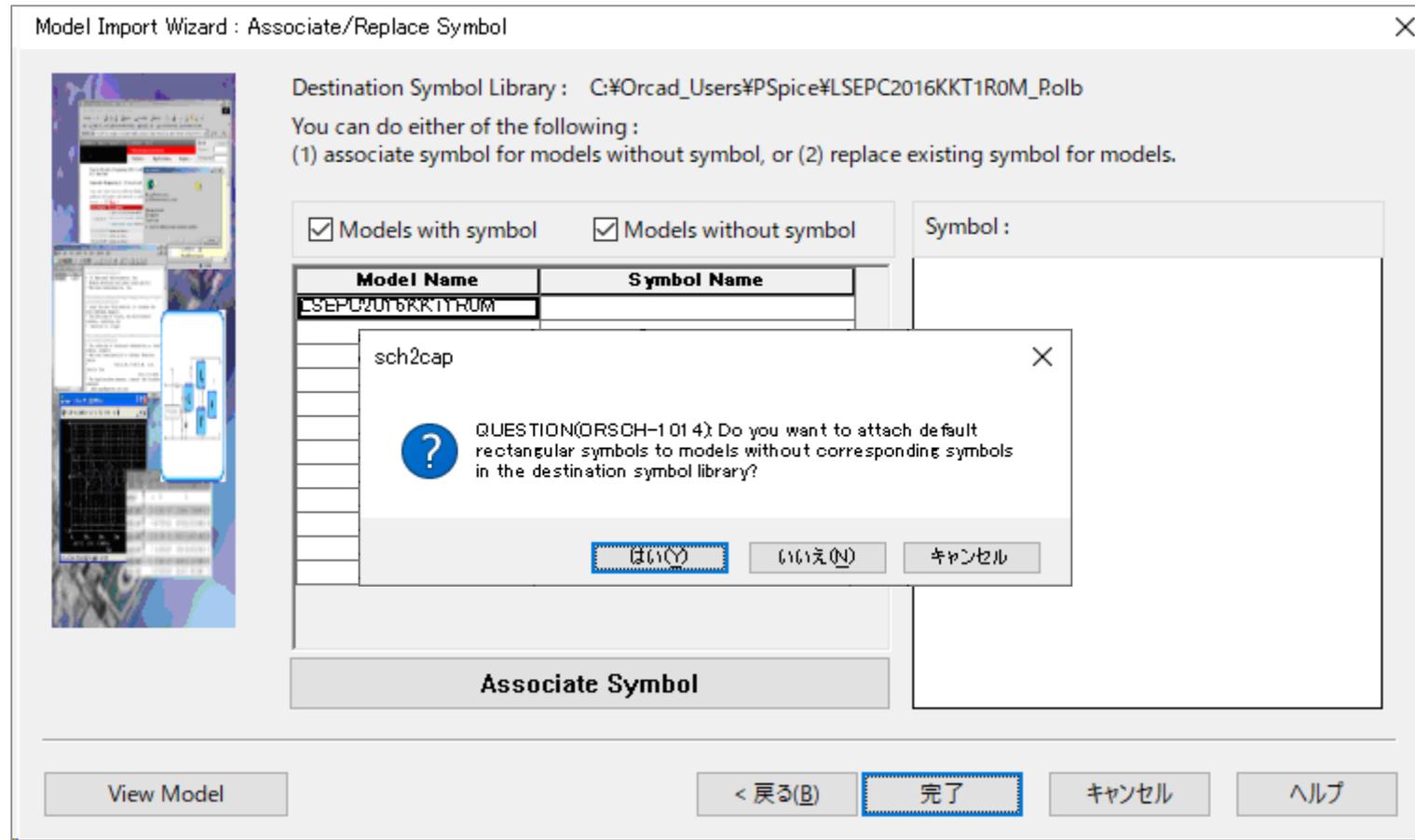
How to use Component Library on OrCAD

Step 2. Select the library on the Model Import Wizard window, then go to next.



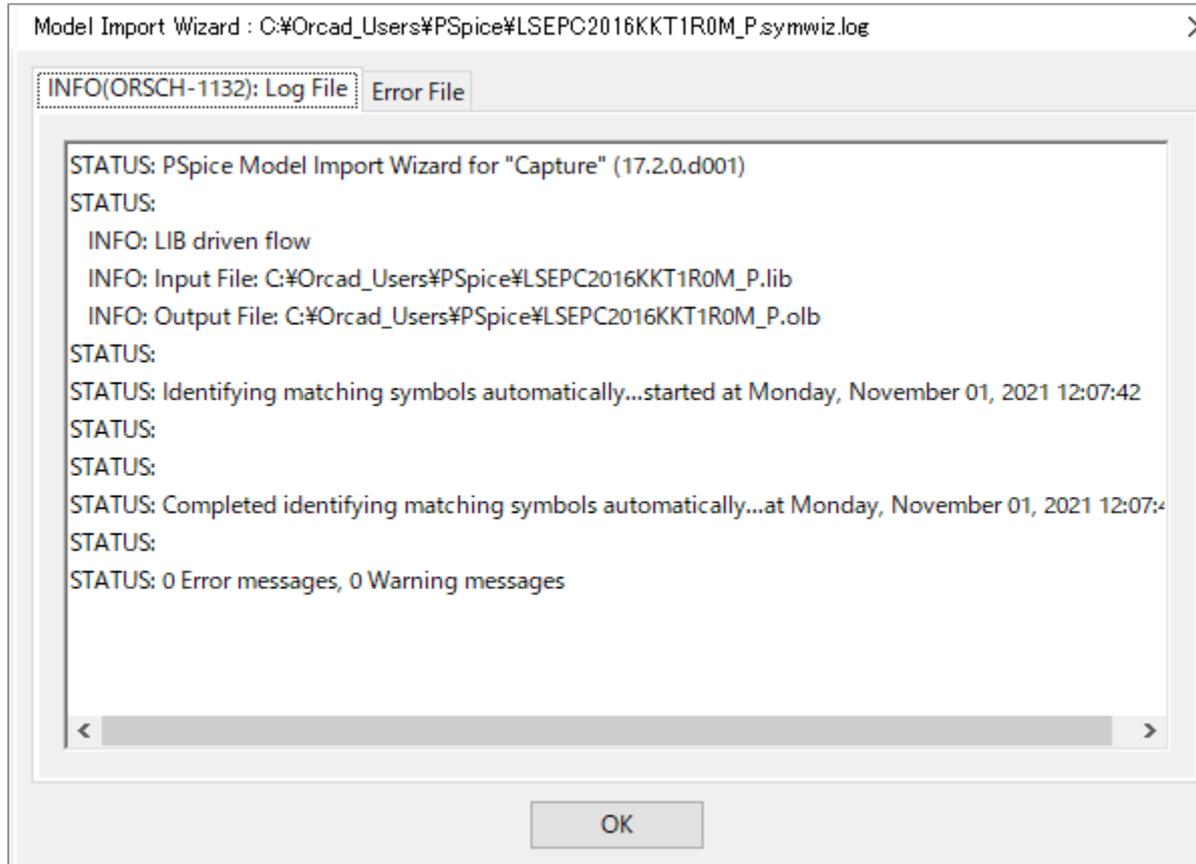
How to use Component Library on OrCAD

Step 3. Click finish button without associating the existing symbols and also click yes on the next dialog.



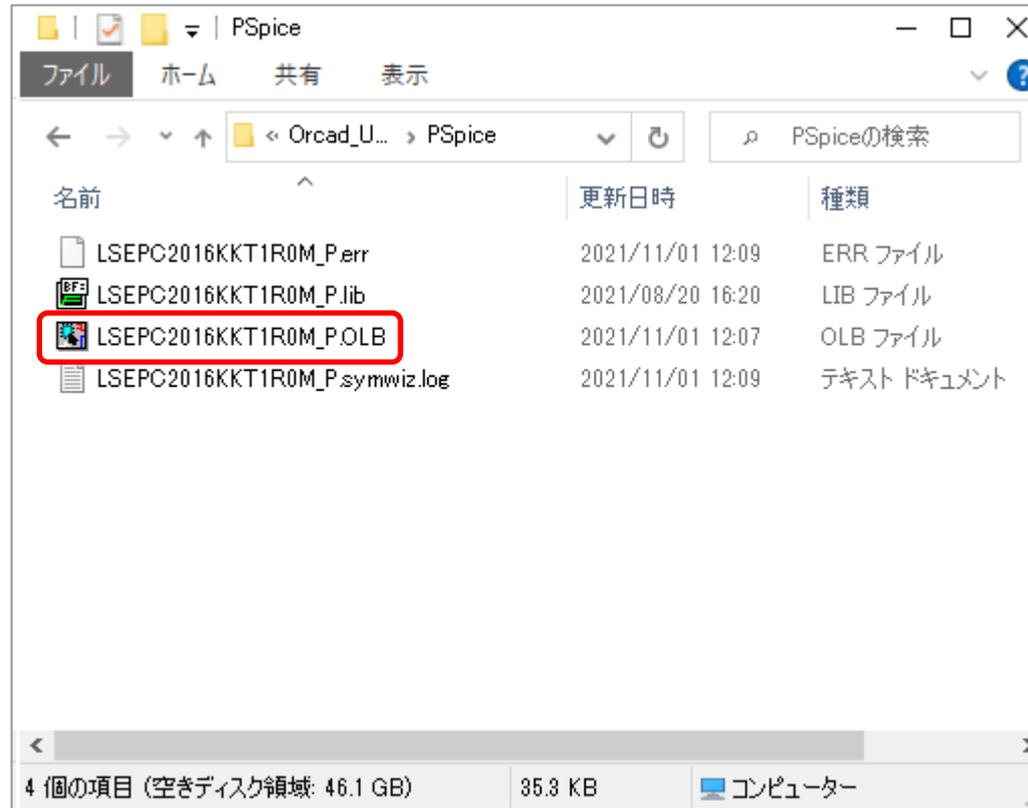
How to use Component Library on OrCAD

Step 4. Click OK on the final log window.



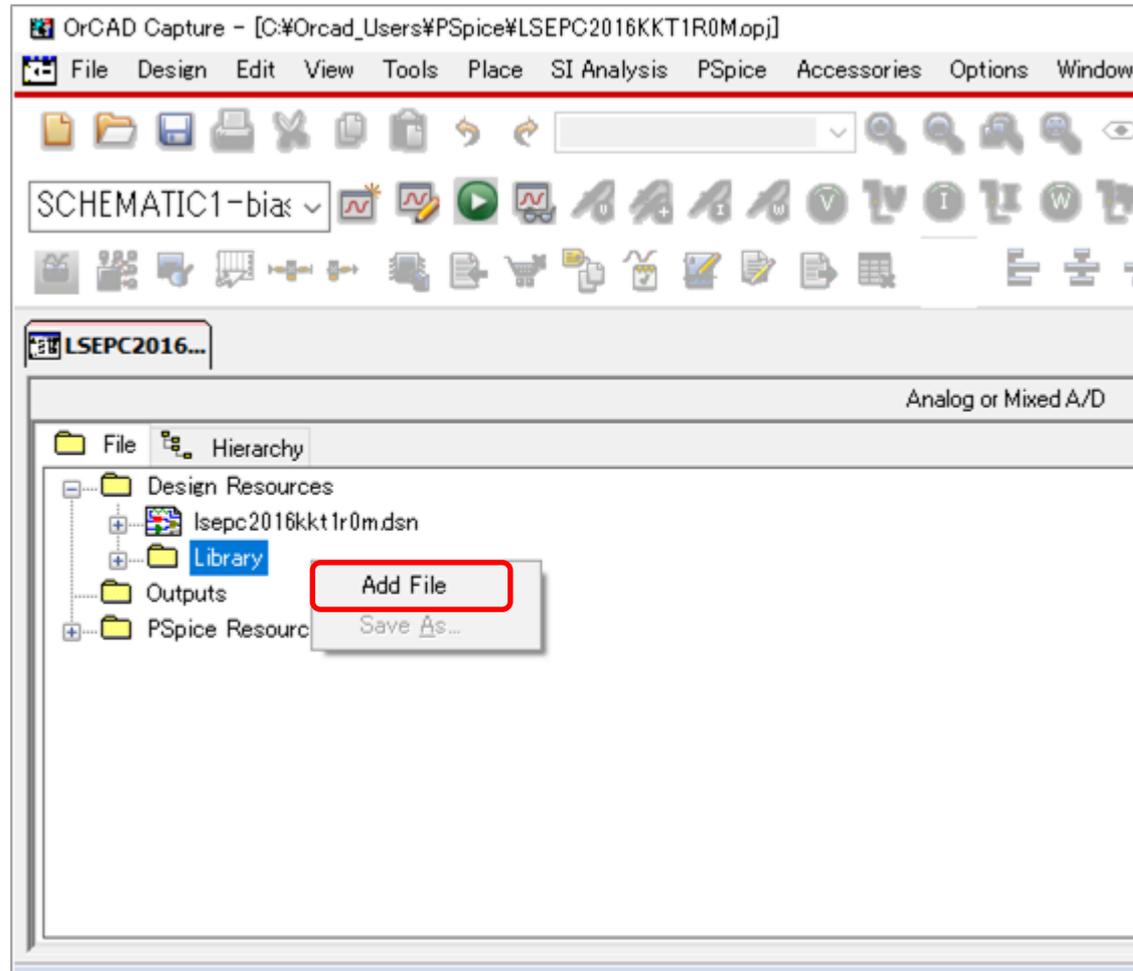
How to use Component Library on OrCAD

Step 5. Verify that the symbol file(.olb) is generated to use on the OrCAD schematic.



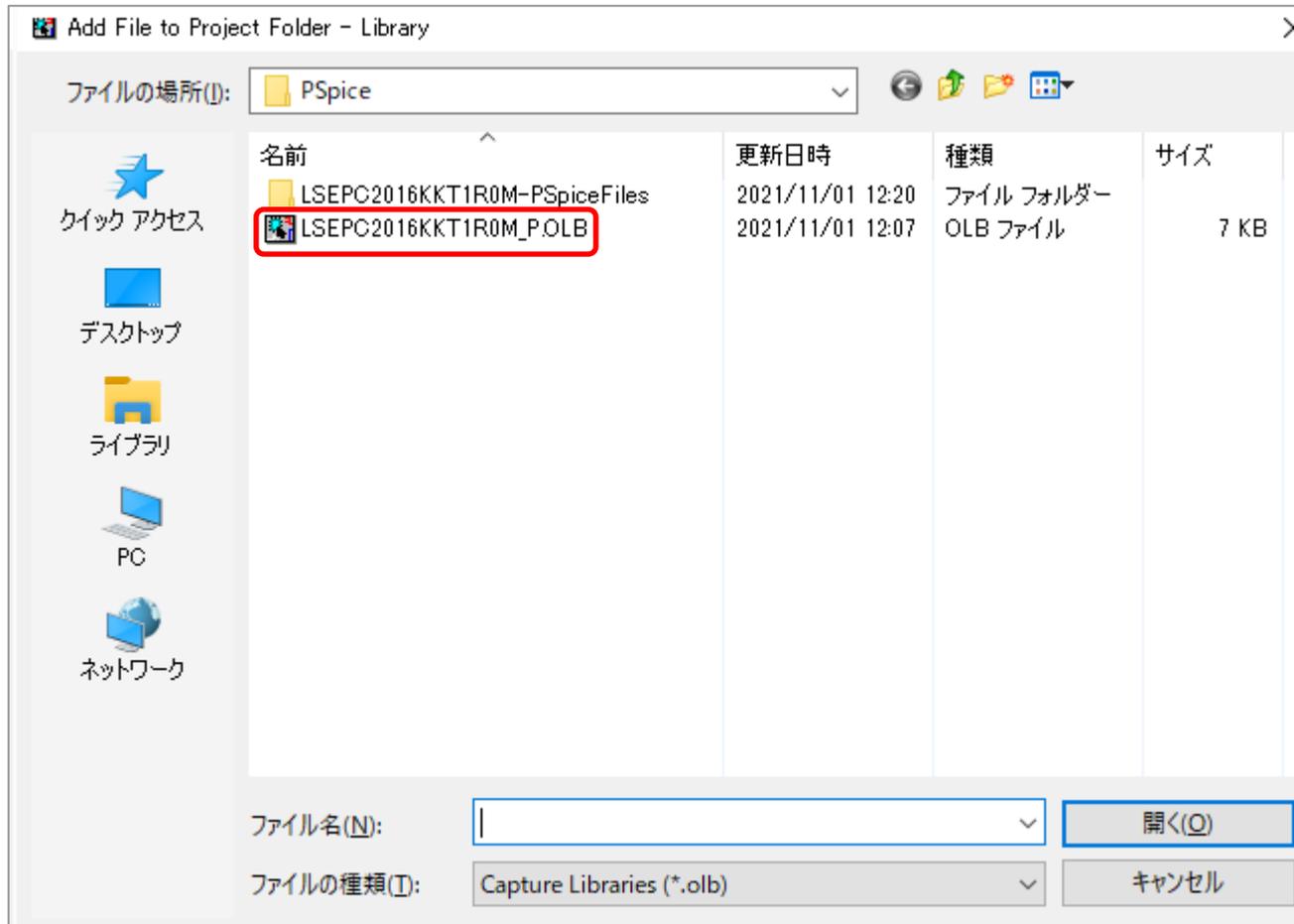
How to use Component Library on OrCAD

Step 6. Launch OrCAD Capture, then right-click Library on the project tree and select Add File.



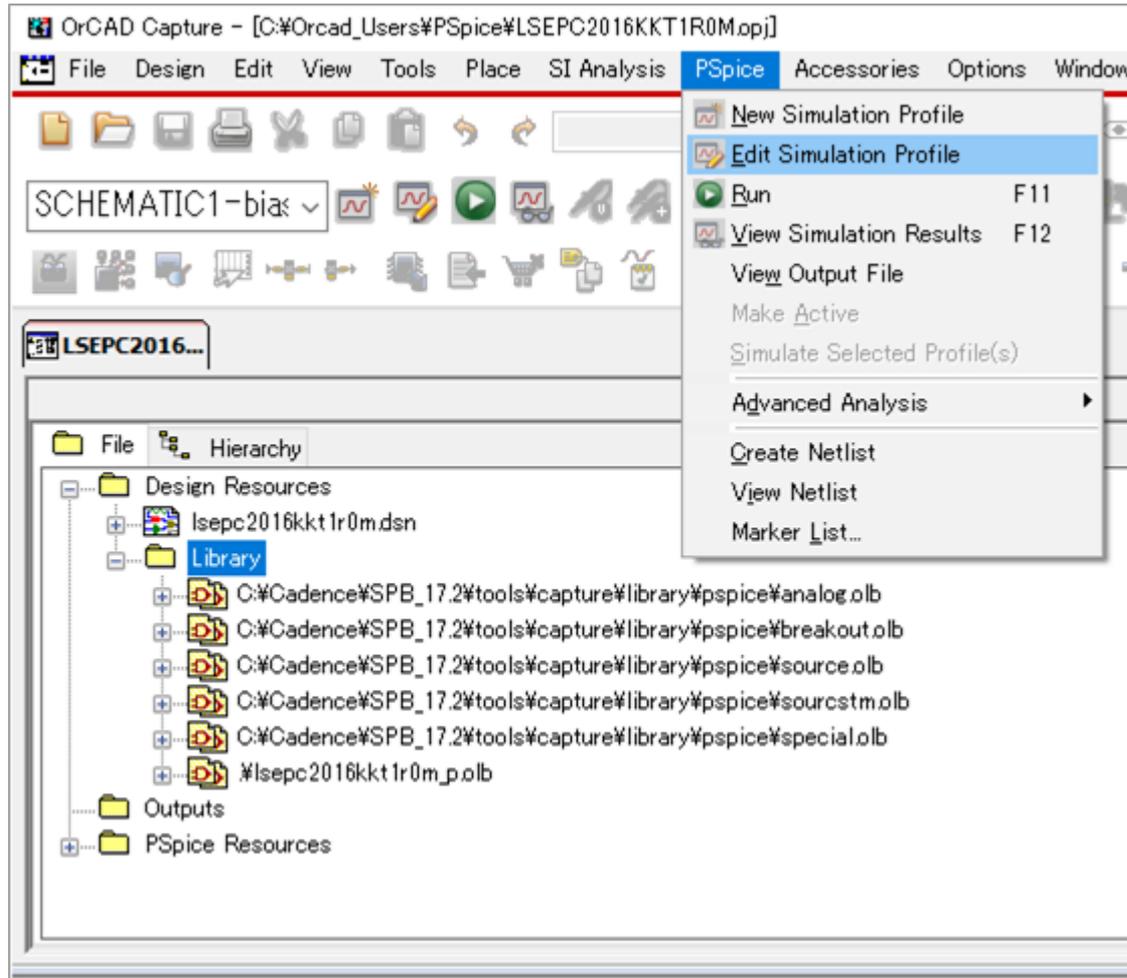
How to use Component Library on OrCAD

Step 7. Select the symbol file(.olb) generated at step 5 to register symbol.



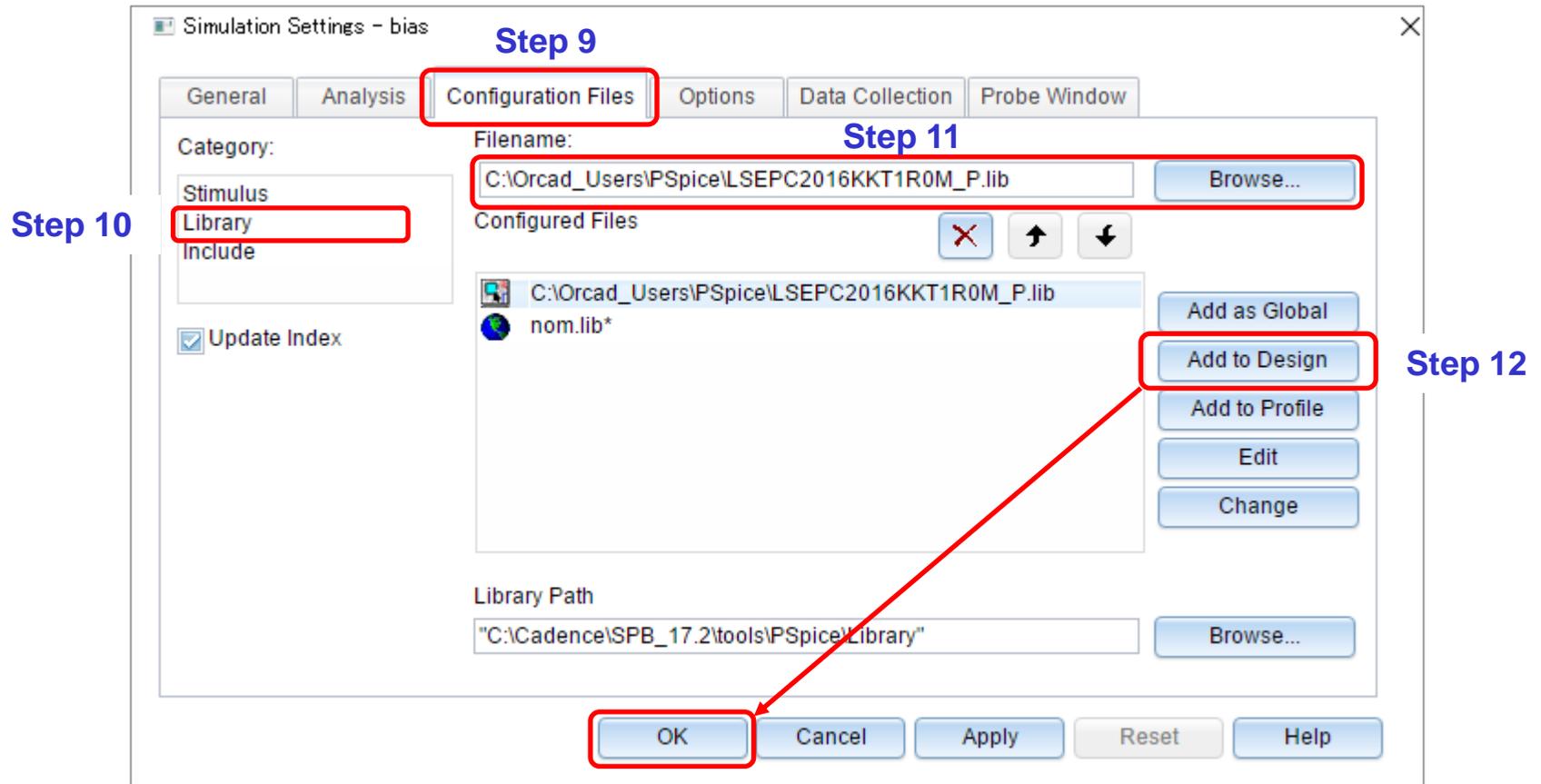
How to use Component Library on OrCAD

Step 8. Select PSpice > Edit Simulation Profile from the menu bar.



How to use Component Library on OrCAD

- Step 9.** Select Configuration Files tab.
- Step 10.** Select “Library” on the Category pane.
- Step 11.** Select the library(.lib) at the Filename section.
- Step 12.** Click Add to Design, then OK to register library(.lib).



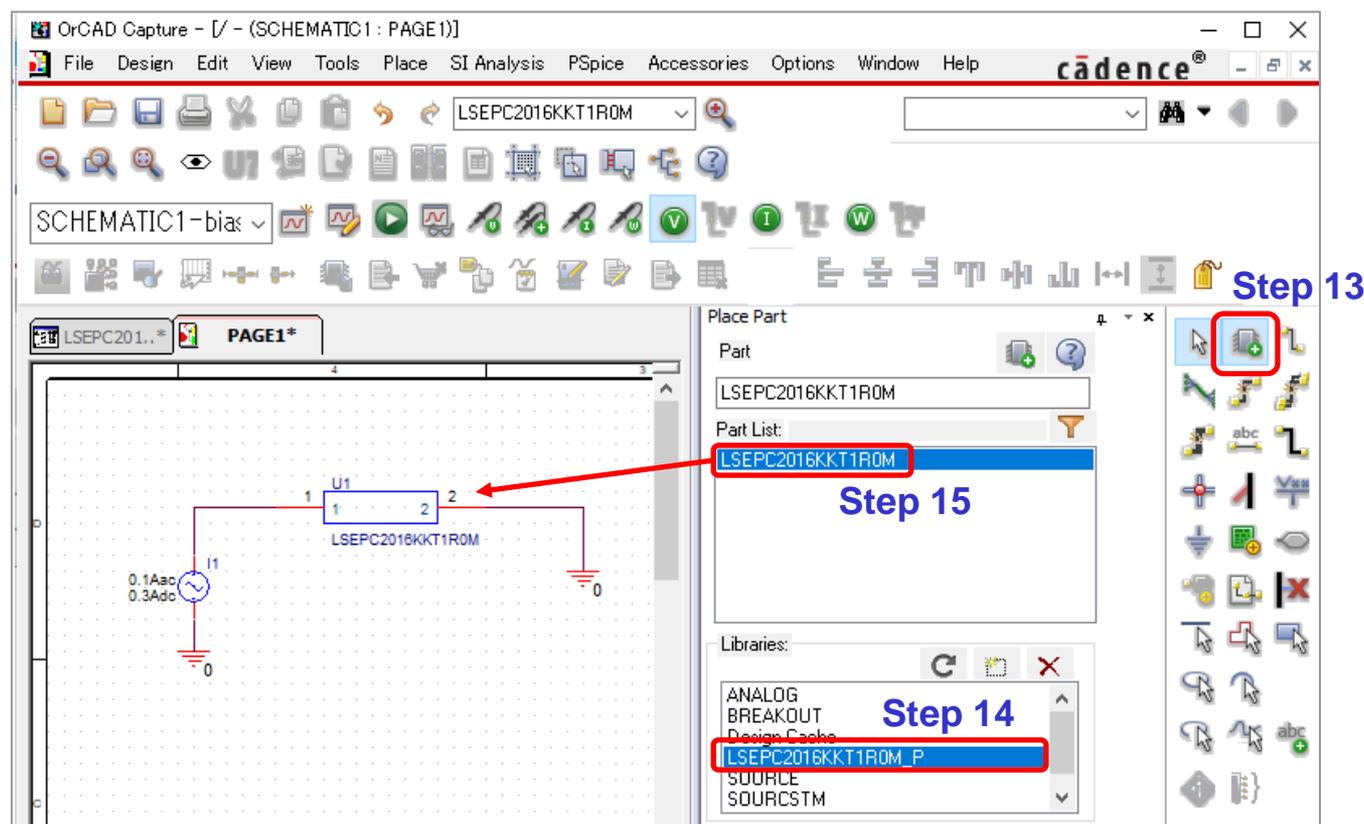
How to use Component Library on OrCAD

Step 13. Open the schematic and select the Part icon.

Step 14. Select the library on the Libraries pane on the Place Part window.

Step 15. Double-click the component on the Part List pane to put on the schematic.

Step 16. Perform the simulation after completing the schematic.



* Refer to page 5 of this manual for setting the ambient temperature of the component.

TAIYO YUDEN